Technology Opportunity

Numerical Simulations of Multifluid Flows

The National Aeronautics and Space Administration seeks to transfer technology and expertise for simulating fluid flows involving multiple fluids or multiple phases.

Potential Commercial Uses

Computational fluid dynamics (CFD) for multifluid flows can be applied in the industrial environment to study flows of multiple or immiscible fluids. It can be used both in research and in design. Applications include

- Fuel atomization
- Paint sprays
- Liquid-metal sprays
- Liquid-jet breakup
- Materials processing
- Casting and molding
- Boiling
- Cavitation

Benefits

- Flows not amenable to experimental investigation can be studied.
- Important parameters can be easily controlled and varied.
- Complete information about the entire flow field is obtained.
- Understanding of the fluid flow is enhanced through simulations and visualization.
- Method can be used in combination with design optimization.
- Method can complement traditional design approaches for equipment and products utilizing flow of gases and/or liquids

The Technology

The power of modern computers can be harnessed to simulate steady state or time evolution of complicated fluid flows by utilizing computational fluid dynamics (CFD). In CFD, the continuous domain of the flow is replaced by a network of points (a grid system). The basic laws of continuum mechanics, namely conservation of mass, momentum, and energy, are used to derive equations that describe the evolution of the flow at each of the points. A computer is then used to solve the system of equations and generate quantitative predictions of the flow field variables, such as pressure, velocity, and temperature. The accuracy of the predictions increases as the density of grid points increases. However, the cost of computing the solution also increases.

Multifluid and multiphase flows pose a special challenge to CFD because of the sharp interfaces between two fluids or two phases. At these interfaces, properties such as density and viscosity can change by many orders of magnitude across the interface and forces due to surface tension act on the fluid. For

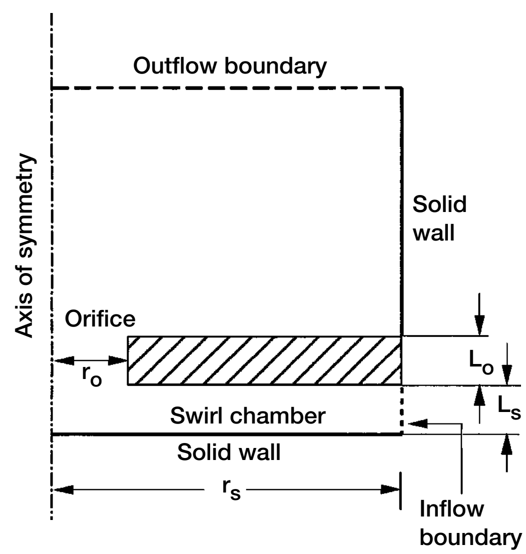

Figure 1.—Schematic of an axisymmetric pressure-swirl atomizer (simplex nozzle). Fuel is injected into the swirl chamber. Because of conservation of angular momentum, the fuel forms a conical liquid sheet as it exits the nozzle through the orifice.

National Aeronautics and Space Administration
Lewis Research Center

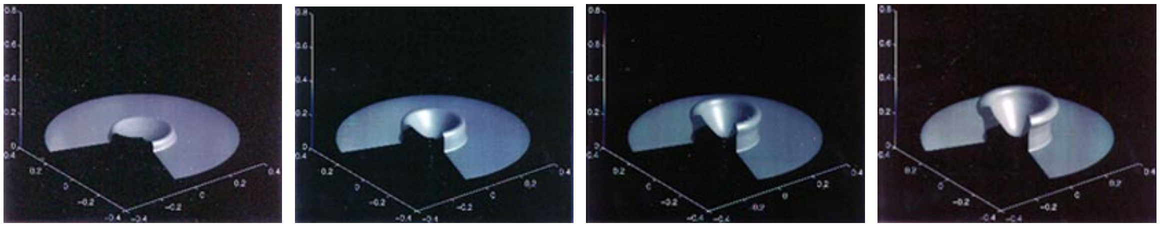

Figure 2.—Formation of a conical liquid sheet ejecting from simplex nozzle.

all practical purposes, the interfaces can be considered a discontinuity in the flow field. Ordinarily, very dense grid systems are required to resolve such discontinuities, making numerical simulations expensive. To reduce the need for very dense grids, special techniques are used to accurately model the flow physics in the neighborhood of the interface.

A special “front tracking method” has been developed to model the behavior of a fluid system at interfaces between two fluids or two phases. In this method, an interface is modeled by a network of nodes that lie on the interface. The nodes are connected by special elements, thus forming a surface or “front” that approximates the shape of the interface. The motion of the interface is simulated by advecting the nodes on the front with the fluid. The motion of the fluid is, in turn, affected by the interface, for example, through surface tension. As the interface moves and changes shape, nodes and elements are adaptively added to the front to ensure adequate resolution at all times. The topology of the front can also change to reflect topological changes in the interface, such as in the breakup of a liquid droplet.

A sample solution of multifluid flow in a pressure swirl atomizer (fig. 1), obtained by using the front tracking method, is shown in figure 2.

Options for Commercialization

Seeking collaboration to develop a general software package targeted at multifluid flow simulation. Seeking to market the expertise and simulation capability on a consulting basis.

Contact

Dr. Erlendur Steinthorsson
Senior Research Associate
Institute for Computational Mechanics
in Propulsion
NASA Lewis Research Center/OAI
22800 Cedar Point Road
Cleveland, OH 44142
Phone: (216) 962-3162
Fax: (216) 962-3200)
E-mail: E.Steinthorsson@lerc.nasa.gov

Key Words

Computational fluid dynamics
Multifluid flows
Multiphase flows

National Aeronautics and
Space Administration
Lewis Research Center